

On numerical modeling of recirculation flows at bay entrances

Dr Miodrag JOVANOVIĆ

University of Belgrade, Serbia
Faculty of Civil Engineering
www.grf.bg.ac.rs/~mjovanovic
mjovanov@grf.bg.ac.rs
mjovanov@eunet.rs

20.09.2018

Abstract

Hydraulic analysis of recirculation flows at bay entrances is presented through a case study of Čukarica Bay on the Sava River in Belgrade, Serbia. Previous studies presented numerical simulations of the recirculation flow using a calibrated two-dimensional (TELEMAC-2D) model and demonstrated how the results were incorporated into a river-training project. The present study extends this analysis by applying the three-dimensional TELEMAC-3D model to provide additional insight into the complex flow structure, with particular emphasis on turbulence characteristics and hydrodynamic conditions relevant to sediment deposition. The results obtained using two turbulence-modelling approaches — a constant eddy-viscosity model and a large-eddy simulation (LES) model — are presented and compared. The study also demonstrates the practical relevance of three-dimensional hydrodynamic modelling for assessing recirculation zones and their potential influence on sediment accumulation at bay entrances. The results indicate that the TELEMAC-2D/3D modelling framework can be a useful tool for evaluating river-engineering solutions aimed at reducing siltation and, consequently, the need for frequent dredging in river bays.

Key words: river bays, recirculation flows, Telemac-2D/3D

1 Introduction

The entrance to the river bay or port basin is a discontinuity of the coast line that causes currents to deviate and the recirculation flow to appear. The resulting current is characterized by a gradual dissipation of kinetic energy, which results in the deposition of the suspended sediment at the entrance to the bay with unfavourable repercussions [10].

There is no possibility to completely prevent sedimentation at bay or port basin entrances. Only measures to slow down the process of sedimentation (i.e. to reduce the frequency of dredging), are to be considered.

Several projects were undertaken in the past by the Faculty of Civil Engineering, University of Belgrade, in order to tackle the problem of sedimentation in the Čukarica Bay, formed by closure of the Sava River branch [6, 7], as shown in Fig. 1.



Figure 1: *Computational domain and bathymetry; the Bay bottom is 8 m above the Sava River thalweg [6].*

One of considered engineering solutions was placement of a low transverse sill at the Bay entrance (Fig. 2), with an idea to at least partially compensate for the missing part of the river coast.

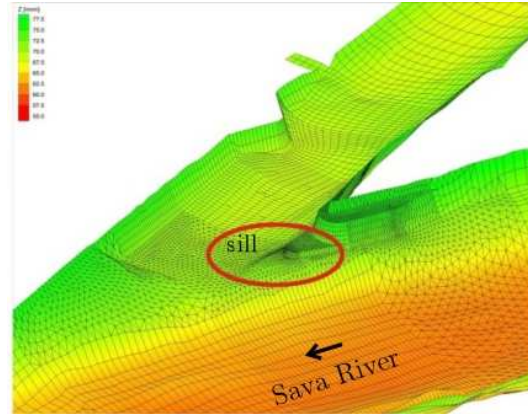


Figure 2: *The river channel before the works and after building a sill at the Bay entrance [6].*

The sill seemed a good choice as the highest sediment concentration is near the bottom of the river bed. The constructive elements of the sill (position, length, top elevation) were optimized according to the results of Telemac-2D numerical simulations. The model calibration was performed by measuring currents in the Sava River by an ADCP device [6, 7].

Since no sediment transport field measurements were available, a direct simulation of sediment accumulation at the Bay entrance could not be reliable, so the analysis was carried out in a qualitative sense (Fig. 3 and Fig. 4). This simplified analysis, which was considered to be sufficient at an engineering level, was based on sediment sampling data (particle size, settling velocity W , etc.).

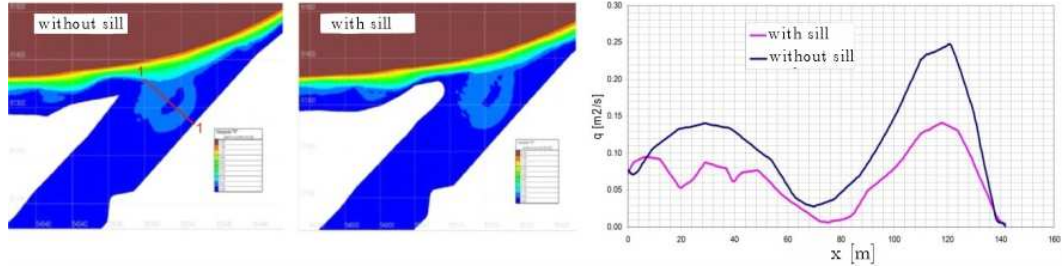


Figure 3: *Results of numerical simulation: distribution of the unit discharge (q [m²/s]) across the recirculation zone for a chosen high river discharge of 3000 m³/s; the minimum discharge in section 1-1 corresponds to the vortex center; to its right is the maximum incoming discharge (from the river into the Bay), while to the left is the return flow towards the river; introduction of the sill results in a reduction of the incoming unit discharge of 44%. Accordingly, less sediment inflow is expected [6].*

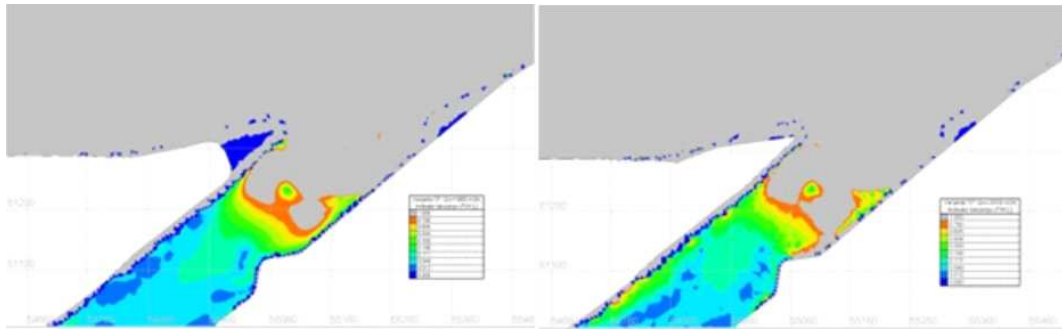


Figure 4: *Results of numerical simulation: values of the ratio of the shear velocity (V_*) to the settling velocity (W) are used for indicating the possibility of suspended sediment settlement; only the region where settlement can be expected, defined by values $V_*/W < 1$ is shown; the size of this region is greater in the case without the sill (diagram on left) than in the presence of the sill (diagram on the right) [6].*

As it was thus concluded that the presence of a transverse sill structure has positive effects on the reduction of sediment deposition at the Bay entrance, the design of the sill was undertaken, and the structure was completed as part of more comprehensive river training works.

After the reconstruction of the Čukarica Bay entrance, the investigation of the recirculation flows was not abandoned. This topic was (albeit with a delay of several years) brought up again with the aim of better understanding such flows through a series of hydraulic parameters that were not used before.

New 2D and 3D computations were carried out with the original geometry (without sill), in the steady flow regime at a high discharge of $3000 \text{ m}^3/\text{s}$. This particular discharge, with a probability of 10% and an average duration of 36 days/year, was chosen as relevant because it completely fills the main river channel, when sediment transport and morphological changes of the river bed are the most expected. The results of the new computations are presented below.

2 Computation of 2D flows

It is well known that the 2D flow in rivers, described by the Saint Venant equations, can be analyzed using various numerical models [5, 6, 7, 9]. In this case the finite element method (FEM) was used, implemented in the Telemac-2D software, with the computational domain discretized as shown in Fig. 5.

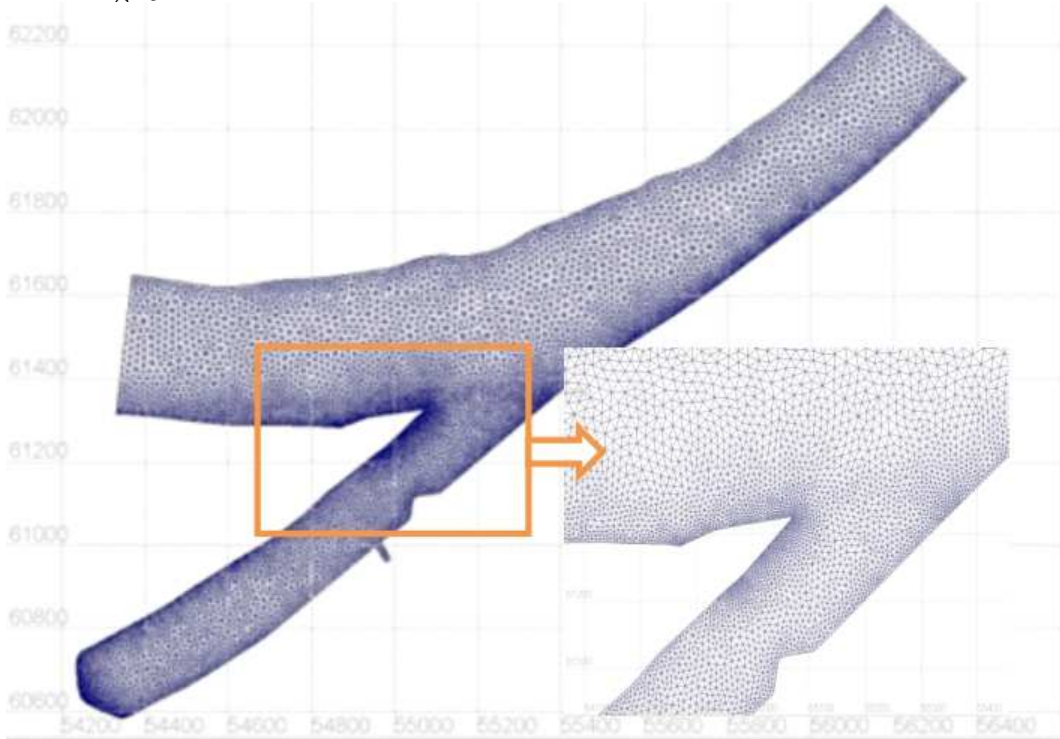


Figure 5: *Computational mesh of the Telemac-2D model, consisting of 16356 triangular elements and 8732 nodes; the average length of elements is 6,5 m, and the average element area, around 50 m^2 [6].*

The numerical solution of the Saint Venant equations, obtained by the fractional-step method, yields an instantaneous distribution of depth and depth-averaged velocity components in the horizontal plane. As the Saint Venant equations are the result of depth averaging of the time-averaged Navier-Stokes (Reynolds) equations, it can be said that the Telemac-2D model belongs to the category of the so-called RANS models ("Reynolds Averaged Navier-Stokes") [2, 3, 5, 9].

One of the objectives of the study was to compare the results obtained by the two approaches for modeling turbulence: one, based on *temporal* averaging (RANS), and second, based on the *spatial* averaging of the Navier Stokes equations (LES). In the first case, the simplest model of constant eddy (turbulent) viscosity was used ("zero equation model", designated as "Model 0") [2, 3, 5, 9]. In the second case, the method of "Large Eddy Simulation" (LES) was used ("LES Model") [1, 4, 8].

At this point, some additional comments on LES may be useful for those who are not too familiar with this topic. Starting from the fact that the turbulent structure of the flow consists of a multitude of temporal and spatial scales (large and small eddies), the LES model separates large eddies that are proportional to the size of the river channel, from small eddies, which form an anisotropic part of the turbulent structure where fluctuations and dissipation of kinetic energy of the *main flow* occur¹.

According to Smagorinskis model [1, 4, 8], the turbulent viscosity (ν_t) is dependent on the size of eddies smaller than the size of the triangular elements of the computational grid:

$$\nu_t = (C_s \cdot \Delta)^2 \sqrt{2 \bar{S}_{ij} \bar{S}_{ij}} \quad (1)$$

where the strain rate tensor:

$$\bar{S}_{ij} = \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (i, j = 1, 2) \quad (2)$$

is defined by depth-averaged velocity $\bar{u}$.

The values of the Smagorinsky constant are subject to estimation and in most cases amount to $C_s = 0,16 \div 0,20$ [4, 8]. It should also be noted

¹The boundary between large and small eddies is defined by the resolution of the computational grid, i.e. the spatial step Δ (in this case, $\Delta = 6,5$ m). For the spatial averaging (filtering) of the basic equations, various "filter functions" are used, where the width of the filter corresponds to the smallest vortex that can be obtained by the solution [1, 4, 8]. Small vortices, whose dimensions are below the computational resolution (abbreviated SGS from "SubGrid Scale") *must be modeled*, because they are not covered by the basic solution. Thus, we talk here about modeling the so-called SGS Reynolds stresses in the dynamic equation, which appear as a consequence of spatial averaging (filtering).

that the Smagorinski model implies that the production of turbulent kinetic energy [5] and the speed of its dissipation are in balance at the level of small eddies.

Velocity vector fields. A comparison of the velocity vector fields at flow rate of $3000 \text{ m}^3/\text{s}$, obtained by the Model 0 and the LES Model, is presented in Fig. 6. The distribution of the velocity intensity in the vortex is shown side by side in the same figure. Streamlines are shown in Fig. 7.

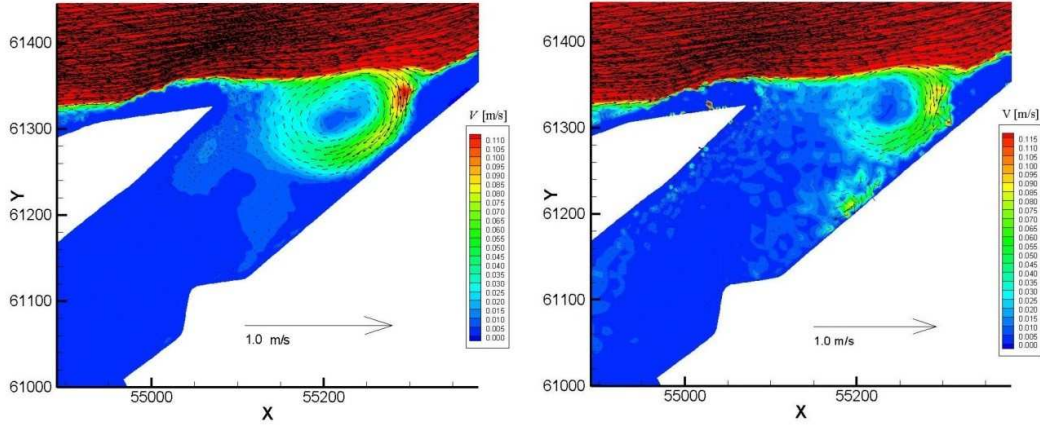


Figure 6: *Velocity vector field and the intensity of velocity; left: Model 0; right: Model LES.*

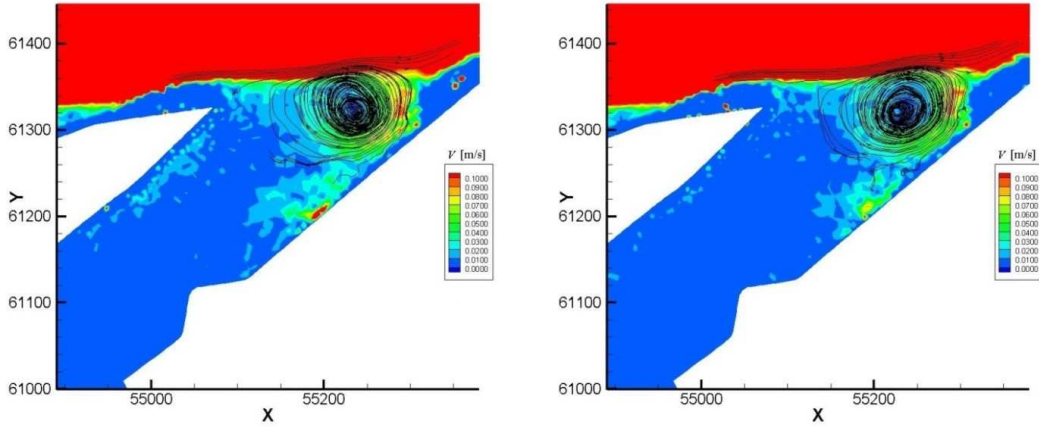


Figure 7: *Streamlines and intensity of velocity; left: Model 0; right: Model LES. Velocities in the recirculation zone are rather low; the highest local speed occurs at the beginning of the Bay entrance and according to both models is about 0,1 m/s. The circular streamline pattern produced by both models is similar.*

Scalar fields. A number of scalar fields can be derived from the velocity

field, as shown in Fig. 8. Examples are the fields of the unit discharge q [m^2/s], shear velocity V_* [m/s], kinetic energy per unit mass KE [$\text{J}/\text{kg} = \text{m}^2/\text{s}^2$] and vorticity (intensity of the velocity rotor) ω [m/s].

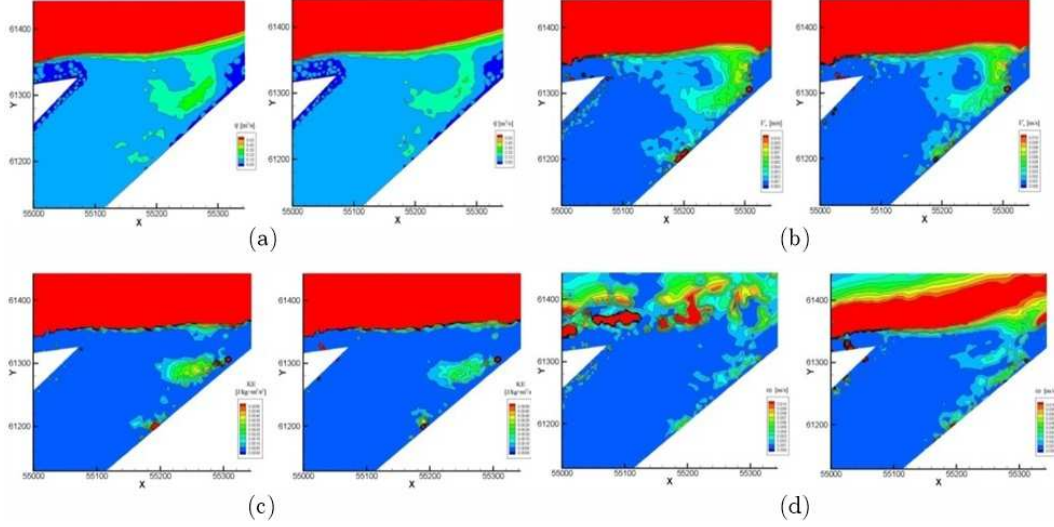


Figure 8: *Scalar fields computed by Model 0 and Model LES; (a) unit discharge; (b) shear velocity; (c) kinetic energy per unit mass, and (d) vorticity.*

The purpose of displaying the intensity and spatial distribution of the given hydraulic quantities is that they are all related to the sedimentation regime and the morphodynamics of the river bed. Unit discharge indicates the intensity of entrainment of water and suspended sediment from the river flow into the Bay. Shear velocity indicates the initiation of entrained sediment as well as the deposition of suspended sediment (as shown in Fig. 4), while the kinetic energy and vorticity are also indicators of the rate of deposition of sediment by recirculation flow. The higher the kinetic energy of the flow per unit mass, the slower the deposition of suspended sediment.

As a measure of the local rotation of fluid particles, vorticity has nothing to do with the global recirculation zone, but refers to the energy status of fluid particles. Namely, if during the movement of fluid particles along a circular path there also occurs their *local rotation*, then the flow contains more energy than in the case when the particles move without local rotation. In this way the vorticity reflects the rate of settling of suspended particles.

The obtained results show that in this case both approaches in turbulence modeling give similar results. One interpretation is that here we deal with a slow recirculation flow, dominantly influenced by friction. This means that turbulence in the horizontal plane plays a secondary role, so various models of turbulence give similar results. This was also observed elsewhere in large rivers. Another possible explanation is that in this case the density of the

spatial filter is too high for the SGS component in the LES Model to be more significant.

3 Computation of 3D flows

As is known in Fluid Mechanics, the spatial (3D) flow is described by the Navier-Stokes equations of conservation of mass and momentum [2, 5, 9]. Again, the FEM model - Telemac-3D was used [9]. The spatial element is in the form of a prism, with a triangular base and vertical sides, with a total of 6 nodes (Fig. 9).

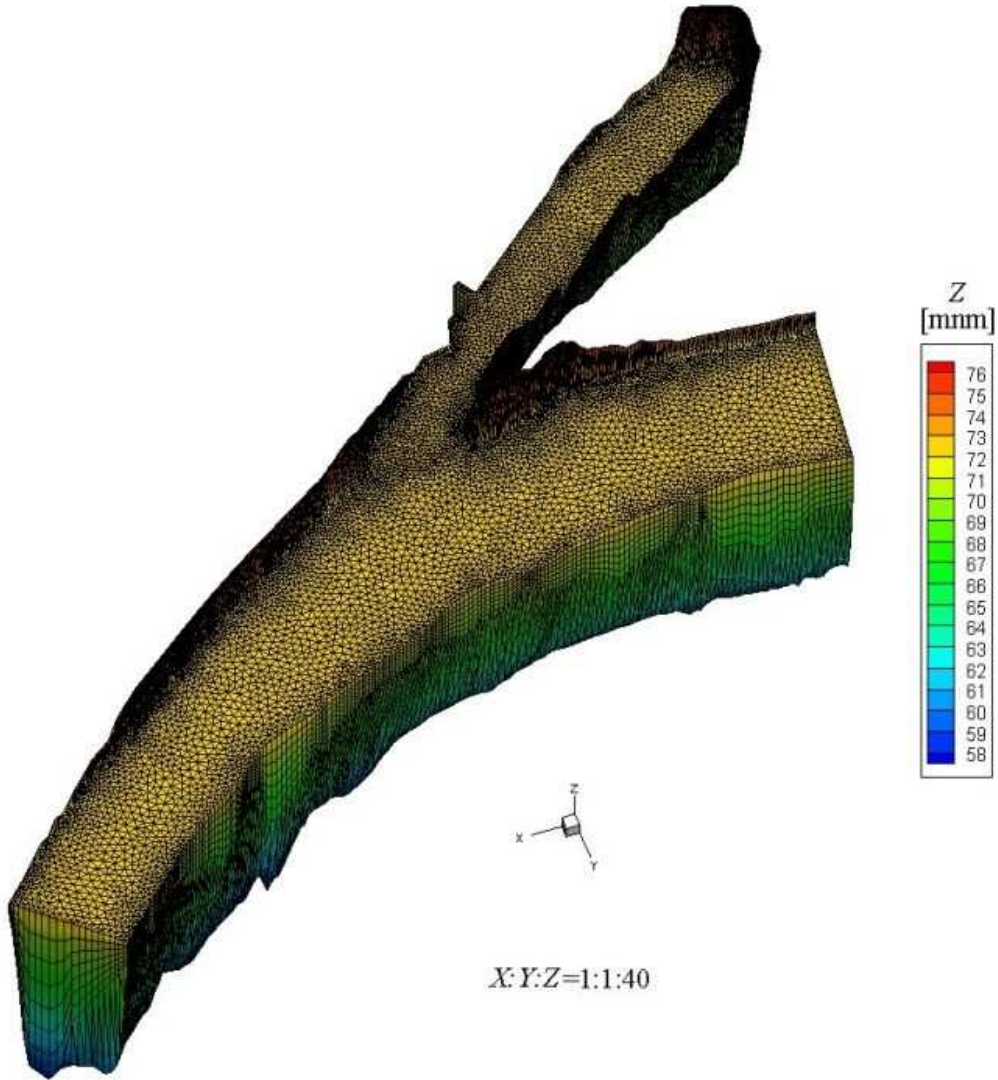


Figure 9: *Computational mesh of the 3D model consists of 8 planes over depth with 8 meshes shown in Fig. 5, thus the 3D model mesh has a total of around 70000 nodes [6].*

The problem of variability of the computational domain in the vertical direction was solved by the "sigma transformation", whereby an irregular physical area is transformed into a fixed computational area with a regular configuration [5, 9].

The basic equations are solved by the fractional step method [5, 9]. Results of numerical simulations of the 3D recirculation flows are presented in Figs. 10–13.

The velocity vectors in Fig. 10 show a velocity decrease along the depth of the recirculation flow. Another, secondary vortex appears, which is not detected by the 2D model. The streamlines indicate the exact position of the core of the recirculation zone.

However, the trajectories of the fluid particles near the core become very irregular, compared to the regular circular trajectories around the perimeter of the recirculation area.

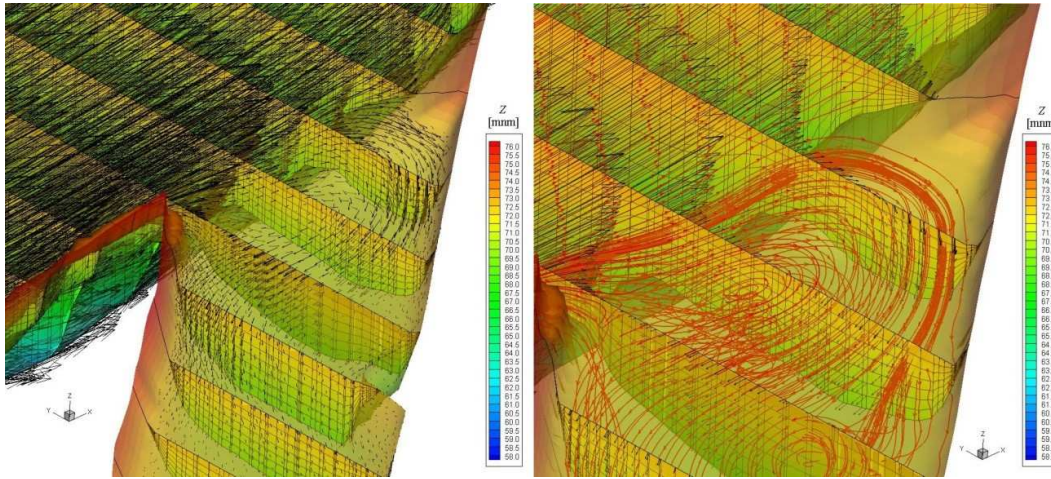


Figure 10: *Numerical simulation of the 3D recirculating flow: velocity vectors and streamlines.*

Fig. 11 shows the advantage of the Telemac-3D model to extract and define currents at any required depth. Fig. 12 illustrates the possibility of studying the change of relevant hydraulic quantities (such as the kinetic energy and vorticity) distribution over the flow depth. Finally, Fig. 13 provides an interpretation of vorticity as a spatial parameter of the current.

4 Two-dimensional vs three-dimensional modeling

The given case study provides a good opportunity to compare the performance of 2D and 3D models. This is done in the following pages.

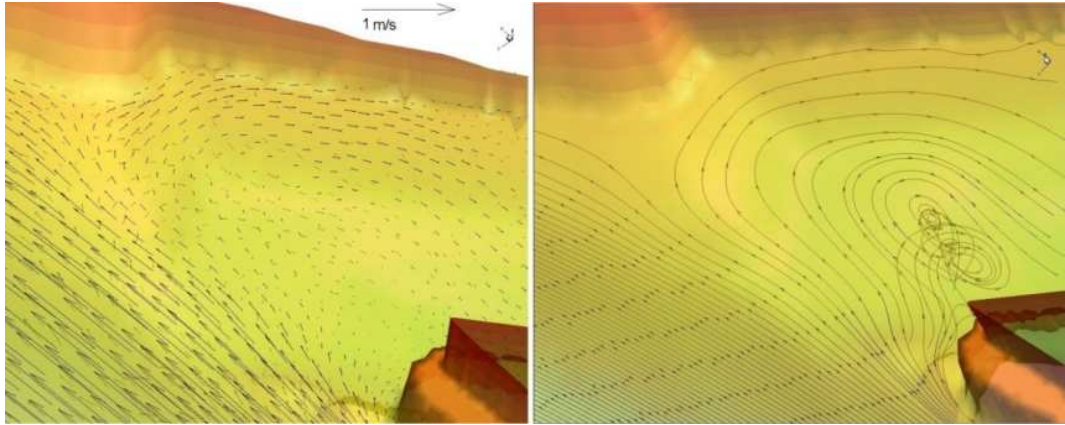


Figure 11: *Numerical simulation of the 3D recirculating flow: velocity and streamlines 1 m below surface.*

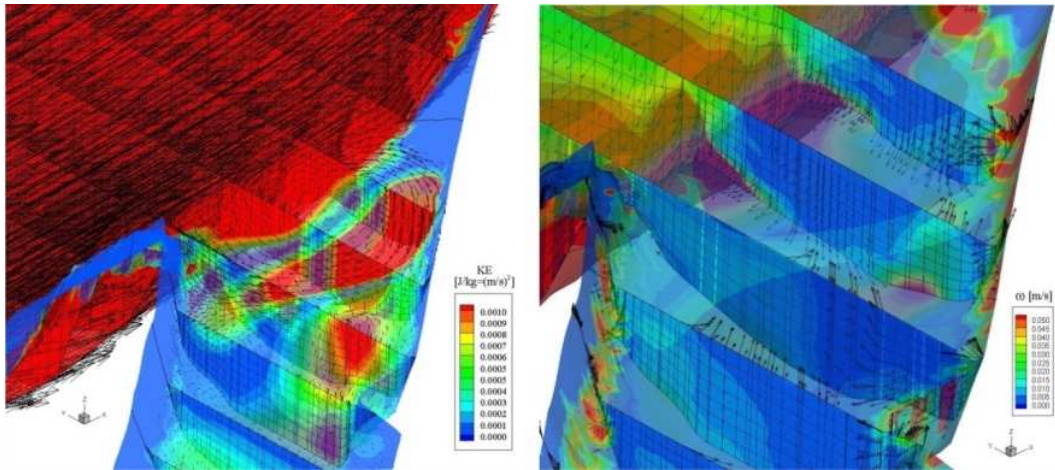


Figure 12: *Numerical simulation of the 3D recirculating flow: kinetic energy and intensity of vorticity.*

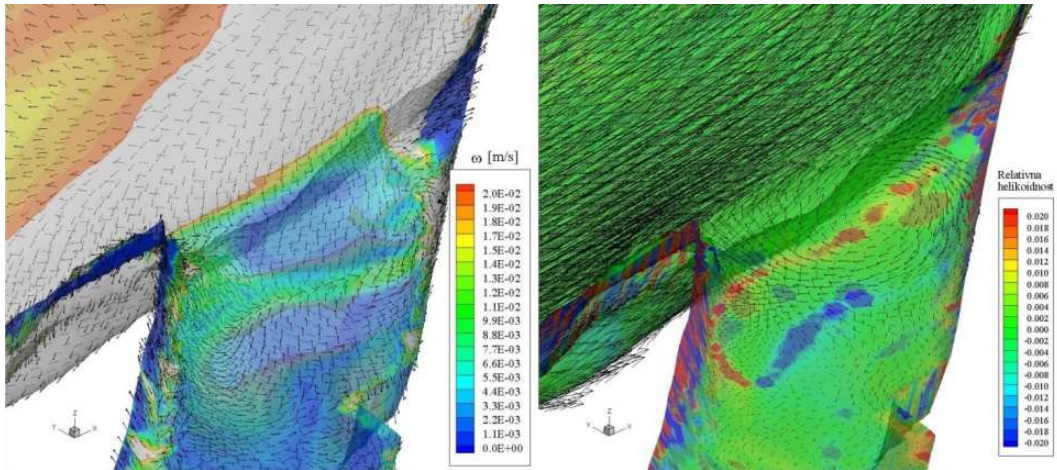


Figure 13: *Vorticity and relative helicity in the recirculating flow.*

By definition, the vorticity of a 3D vector field is a rotor of velocity $V(u, v, w)$:

$$\vec{\omega} = \text{rot}\vec{V} = \nabla \times \vec{V} = \begin{bmatrix} \vec{i} & \vec{j} & 0 \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ u & v & w \end{bmatrix} \quad (3)$$

Vorticity vector field with vorticity intensity and helicoidity is shown in Fig. 13.

As already stated, vorticity can be regarded as an indicator of the rotation of each fluid particle. The intensity of vorticity corresponds to twice the angular velocity (the "right-hand rule" is used for the direction of rotation). As velocity vectors are tangent to streamlines, vorticity vectors are tangent to vortex lines. Unlike streamlines, which describe the direction of fluid motion, vortex lines describe the direction of the local rotation of fluid particles. Vortex lines are defined by the direction of the vorticity vector and represent the local axes of rotation of fluid particles. A family of vortex lines passing through a closed curve forms a vortex tube. According to Stokes theorem, the circulation around an arbitrary closed contour is equal to the flux of vorticity through the surface bounded by that contour. Consequently, circulation can be interpreted as a measure of the strength of a vortex tube and can be used as an indicator of the intensity of vortical flow structures².

An additional indicator of the 3D field can be the scalar product of the velocity vector and the vorticity vector. This scalar quantity is called "helicity". The physical meaning of a helicity becomes clearer when it is normalized and displayed as "relative helicity":

$$H_r = \frac{\vec{\omega} \cdot \vec{V}}{|\omega||V|} = \cos \alpha_{\omega,V}. \quad (4)$$

It can be seen that this quantity represents the cosine of the angle between the velocity and vorticity vectors, so that the values of H_r range between -1 and +1. The relative helicoidity is an indicator of the orientation of the velocity vector field in relation to the vorticity vector field. The closer the value of H_r is to 1, the more the vectors $\vec{\omega}$ and $\vec{V}$ tend to be parallel. This happens near the core of eddies directed in the direction of the main flow (for example eddies left behind by ships).

Having this in mind, the distribution of the H_r variable provides an effective way of locating the predominantly spatial structure of the recirculating flow. If the velocity vector field is superimposed over the scalar field H_r

²The subject of vorticity and circulation has been covered in a large number of articles and textbooks in fluid mechanics, and the physical interpretation of these concepts is available through filmed experiments, for example: <http://web.mit.edu/hml/ncfmf/09VOR.pdf>.

(Fig. 13), it becomes possible to detect where the flow has a pronounced spatial (3D) character, and where it can be considered to be planar (2D). In this particular case, it is observed that the relative helicity values are small, which means that the vectors $\vec{\omega}$ and $\vec{V}$ are mutually orthogonal, which leads to the conclusion that the given recirculating flow can be well numerically simulated by means of a 2D model, and that the 3D model is not necessary. The stated statement raises the question of whether this is completely true and how much differ all considered results of the two used models?

To answer this question, the velocities obtained by the Telemac-3D model were depth-averaged³. In the same way were depth averaged all other relevant variables. These results were compared with the results obtained by the "true" Telemac-2D model. The result of this comparison are presented in Figs. 14 and 15. All calculations were made for a discharge of 3000 m³/s.

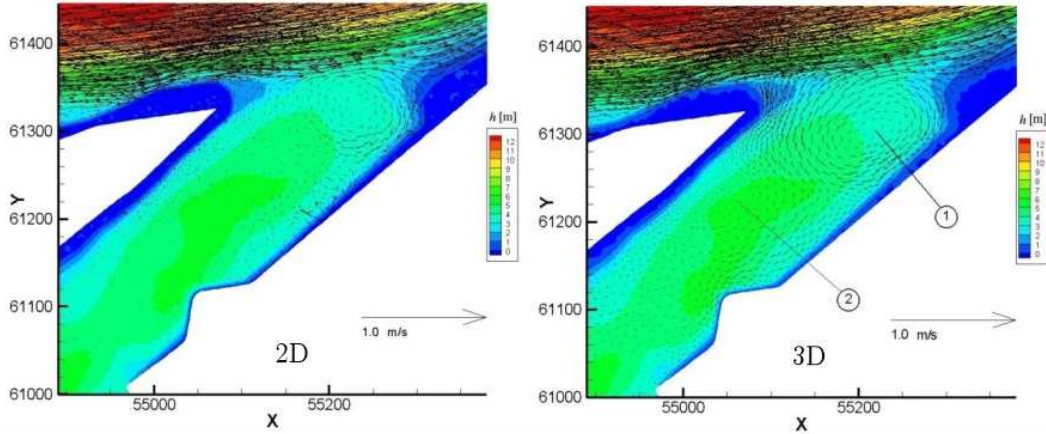


Figure 14: *Comparison of vector velocity fields obtained by the 2D and the 3D model; the numerical values do not differ much, but the recirculation zone provided by the 3D model is noticeably larger, and in addition to the primary vortex, another – secondary vortex rotating in the opposite direction, can be detected.*

The size and shape of the recirculation zone is compared in Fig. 15. It can be seen that, in contrast to that of the 2D model, the recirculation zone of the 3D model practically spans the entire width of the Bay, and spreads more towards the interior of the Bay.

These results somewhat relativize the earlier conclusion (derived from the relative helicity) that the spatial effects of the given recirculating flow are relatively weak, implying that the 2D model is quite sufficient for the numerical simulation in this case study.

³Depth averaging of the velocity components was performed by depth integration using the trapezoidal rule based on the values calculated in the 8 computational planes of the 3D model.

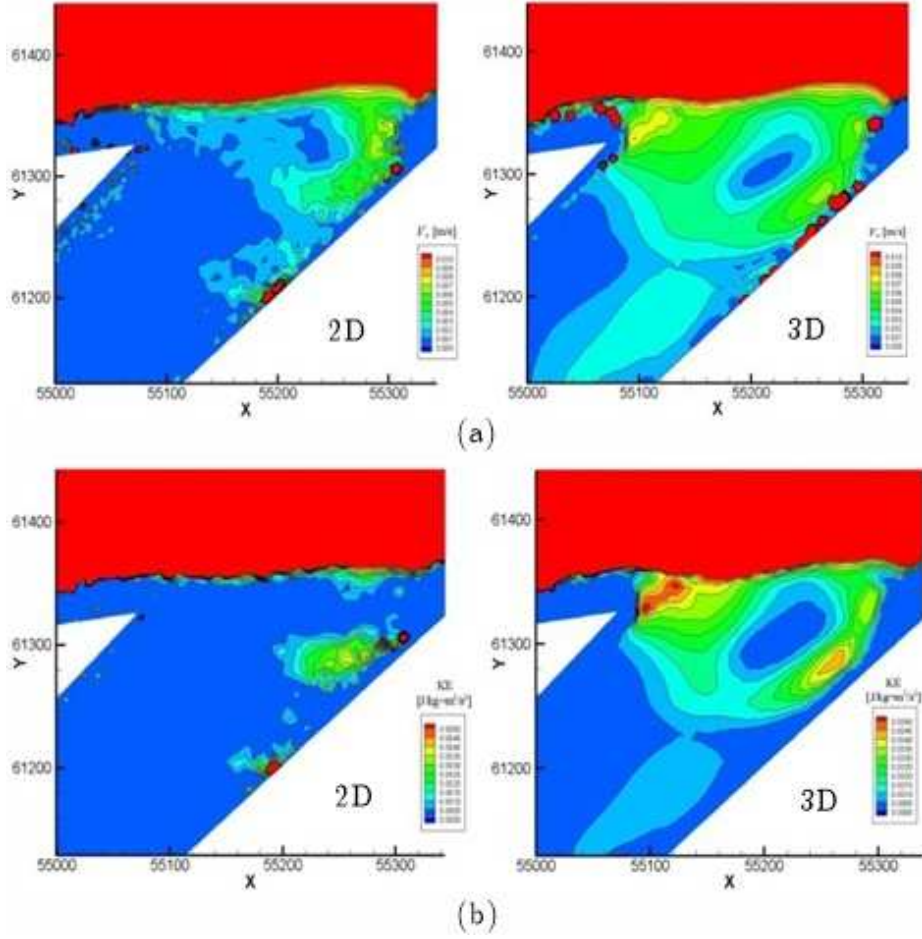


Figure 15: *Comparison of scalar fields obtained by the 2D and 3D models: (a) shear velocity; (b) kinetic energy per unit mass.*

5 Conclusions

1. Telemac-2D/3D modelling framework is a powerful tool for solving practical problems involving complex hydraulic and morphodynamic processes.
2. In numerical simulation of large river flows, it is not a rare case that the results are not greatly influenced by the applied turbulence model, because the current is primarily determined by the channel geometry and friction.
3. In computational grids with a large number of small elements, the density of the spatial filter may be too high for the SGS component to be expressed in the LES model, so very similar results can be obtained with a simple turbulence model with constant eddy viscosity.

4. Vorticity and helicity can be useful indicators of the character of recirculation flows, pointing to regions of increased energy of rotating fluid particles, as well as regions of highly spatial (3D) flow. Both indicators are important for evaluation of the silting potential in bays.
5. The advantage of the 3D model (compared to the 2D model) is not only that it provides distribution of hydraulic variables over the flow depth, but also in a more accurate distribution of those variables in the horizontal plane. Thus, Telemac-3D model can be highly recommended for analysis of the recirculation flows.

References

- [1] Bourgoïn, A., Guillou, S., Thiébot, J., Benhamadouche, S., Ata, R. (2017) *Development of a Large-Eddy-Simulation for Free Surface Complex Flows*, 37th IAHR World Congress.
- [2] Chung, T.J. (2002) *Computational Fluid Dynamics*, Cambridge University Press.
- [3] Ferziger, J.H., Peric, M. (1996) *Computational Methods for Fluid Dynamics*, Springer, Berlin, Heidelberg, New York.
- [4] Germano, M., Piomelli, U., Moin, P., Cabot, W. H. (1991) *Dynamic Subgrid-Scale Eddy Viscosity Model*, Physics of Fluids A, 3(7), 1760-1765.
- [5] Hervouet, J-M. (2007) *Hydrodynamics of Free Surface Flows - modelling with the finite element method*, John Wiley & Sons.
- [6] Jovanovic, M. et al. (2011) *Hydraulic Study of the Entrance to Čukarica Bay Following the Construction of the New Bridge over the Sava River* (in Serbian), University of Belgrade, Faculty of Civil Engineering.
- [7] Jovanovic, M. et al. (2006) *Upgrading Environmental Projects by CFD Modelling*, 23. Conference of the Danube Countries on the Hydrologic Forecasting and Hydrological Bases of Water Management, Belgrade, Serbia.
- [8] Smagorinsky, J. (1963) *General Circulation Experiments with the Primitive Equations: I. The Basic Experiment*, Monthly Weather Review, 91(3), 99-164.
- [9] Telemac modelling system (2010), Operating and Reference Manuals.
- [10] Van Schijndel, S., Kranenburg, C. (1998) *Reducing the siltation of a river harbour*, Journal of Hydraulic Research, Vol. 36, No. 5, pp. 803-814.